



LAWRENCE  
LIVERMORE  
NATIONAL  
LABORATORY

# Inhibition of Turbulence in ICF Hot Spots by Viscous Dissipation

C. R. Weber, D. S. Clark, A. W. Cook, L. E.  
Busby, H. F. Robey

January 14, 2014

Physical Review E

## **Disclaimer**

---

This document was prepared as an account of work sponsored by an agency of the United States government. Neither the United States government nor Lawrence Livermore National Security, LLC, nor any of their employees makes any warranty, expressed or implied, or assumes any legal liability or responsibility for the accuracy, completeness, or usefulness of any information, apparatus, product, or process disclosed, or represents that its use would not infringe privately owned rights. Reference herein to any specific commercial product, process, or service by trade name, trademark, manufacturer, or otherwise does not necessarily constitute or imply its endorsement, recommendation, or favoring by the United States government or Lawrence Livermore National Security, LLC. The views and opinions of authors expressed herein do not necessarily state or reflect those of the United States government or Lawrence Livermore National Security, LLC, and shall not be used for advertising or product endorsement purposes.

# Inhibition of Turbulence in ICF Hot Spots by Viscous Dissipation

C. R. Weber,\* D. S. Clark, A. W. Cook, L. E. Busby, and H. F. Robey

*Lawrence Livermore National Laboratory, Livermore, California 94550, USA*

(Dated: January 8, 2014)

Achieving ignition in Inertial Confinement Fusion (ICF) requires the formation of a high temperature ( $>10$  keV) central hot spot. Turbulence has been suggested as a mechanism for degrading the hot spot conditions by altering transport properties, introducing colder, mixed material, or reducing the conversion of radially-directed kinetic energy to hot-spot heating. We show, however, that the hot spot is very viscous and the assumption of turbulent conditions in the hot spot is incorrect. This Letter presents the first high-resolution, three-dimensional simulations of National Ignition Facility (NIF) implosion experiments using detailed knowledge of implosion dynamics and instability seeds and including an accurate model of physical viscosity. We find that when viscous effects are neglected, the hot spot can exhibit a turbulent kinetic energy cascade. Viscous effects, however, are significant and strongly damp small-scale velocity structures, with a hot spot Reynolds number in the range of only  $10 - 100$ .

PACS numbers: 52.57.-z, 52.35.Ra, 51.20.+d

Experiments at the National Ignition Facility (NIF) [1] seek to compress a capsule (1.11 mm radius), consisting of deuterium-tritium (DT) fuel and an outer plastic (CH) ablator, to sufficient temperatures and densities that a self-sustaining thermonuclear burn is achieved. At the time of ignition, the fuel should be assembled into a low density ( $\sim 100$  g/cm<sup>3</sup>), high temperature ( $>10$  keV) central hot spot surrounded by high density ( $\sim 1000$  g/cm<sup>3</sup>) DT fuel. To achieve this goal of Inertial Confinement Fusion (ICF), NIF is using an indirect drive configuration [2], where 192 laser beams irradiate the inner wall of a high-Z hohlraum, surrounding the centrally-located capsule with a bath of x-ray radiation at a temperature of up to 300 eV. The outer layers of the capsule ablate and launch a series of shocks inward. Perturbations on capsule surfaces can grow in amplitude and, if sufficiently large, could break up the ablator or mix higher Z material with DT fuel, and thereby cool the central hot spot. Greater-than-expected perturbation growth is one of the possible reasons that NIF experiments have yet to achieve ignition [7, 8].

The most comprehensive understanding of a particular experiment comes through detailed post-shot modeling [9] using the radiation hydrodynamics code HYDRA [10]. This modeling effort incorporates a wide range of the actual conditions of a particular experiment, including capsule metrology, surface roughness, and radiation drive asymmetry. The strength of the radiation drive is tuned to match the measured shock velocity history [11], the capsule implosion dynamics [12], the bang time (the time of peak neutron production), and the column density ( $\rho R$ ) [13] from several specialized experiments. For some implosions, post-shot modeling can produce good agreement with the measured hot spot size, temperature, and the column density of the fuel, yet the neutron yield is over-predicted by a factor of several ( $\sim 2-3\times$ ) [9, 14]. In some poor performing experiments, the discrepancy with the simulation is much larger ( $>10\times$ ) and measure-

ments show [7, 8] that ablator material is mixing into the hot spot in much greater amounts than the simulations predict.

These discrepancies raise the question of what effects are being left out of the modeling effort. To date, full-sphere 3D simulations have included perturbations up to Legendre mode 100. The computational expense of these simulations currently precludes higher-mode 3D simulations, although it is expected that this range includes the most significantly growing modes [15]. Even with the current set of perturbation modes, one can ask if higher resolution or a different hydrodynamics code would produce different results. This would be the case if, for example, numerical dissipation were removing some of the kinetic energy that higher resolutions or higher fidelity numerics would preserve. Recent simulations [16] observed that increased resolution and 3D effects lead to a turbulent kinetic energy spectrum in a model ICF problem with mode 30 perturbations. Turbulent mixing could cool the hot spot through enhanced conduction from turbulent transport or through increased radiative losses by mixing ablator material. A turbulent flowfield could also contain energy that has not converted into internal energy. Indeed static thermodynamic modeling of NIF implosions suggest that  $\sim 3$  kJ of energy remains as residual kinetic energy for some experiments [17]. This is larger than simulations predict ( $\sim 1$  kJ) [9] and larger than bulk velocity measurements can explain [18], thus leaving the possibility of turbulent flows.

With this motivation, this Letter presents 3D capsule simulations using a different hydrodynamics code, Miranda [19, 20], with conditions that adhere as closely as possible to the those set forth in the HYDRA post-shot modeling effort [9]. The Miranda code was chosen to elucidate the effects of turbulence and mixing, as the code uses 10th-order spatial accuracy with a fixed Eulerian mesh and employs Large Eddy Simulation (LES) modeling of sub-grid scale turbulent dissipation. Importantly,

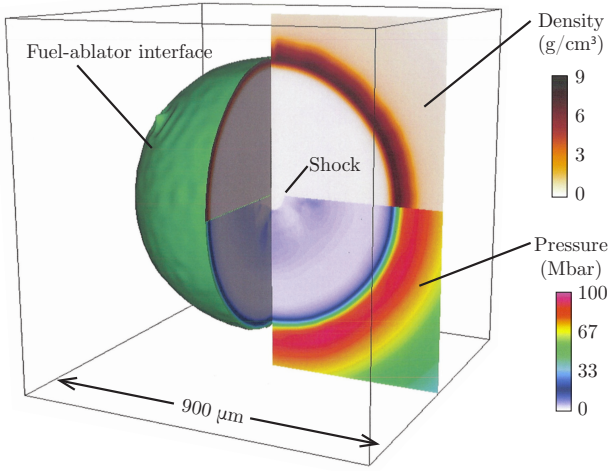


FIG. 1. Simulation setup in Miranda showing the  $900\ \mu\text{m}$  extent of the domain, the fuel-ablator interface of the capsule and slices of density and pressure.

Miranda includes models of physical diffusion and viscosity, of which the latter was found crucial in accurately capturing the hot spot dynamics. The Miranda cases discussed here, in contrast to HYDRA, did not include radiation transfer or thermonuclear burn.

The early stage of the implosion is necessarily simulated using HYDRA, as multigroup radiation diffusion, Lagrangian grid motion, and spherical coordinates are important in resolving the ablation front dynamics. Therefore the Miranda simulation is initialized from the results of the HYDRA simulation once the shocks have broken out of the shell and the perturbations have grown in amplitude but remain linear [21]. At this time the material, density, ion temperature, and velocity data in the central  $(900\ \mu\text{m})^3$  box of the HYDRA simulation are imported into Miranda to start the simulation. As shown in Fig. 1, at this time (21.66 ns) the shell has compressed to  $340\ \mu\text{m}$ , the shock has nearly reached the center, and the perturbations are on the order of  $1\ \mu\text{m}$  in amplitude. In this example, the HYDRA simulation was modeling NIF shot N120205 and included the measured outer surface perturbations up to mode 50, the observed DT fuel surface defects, and measured, broad-mode power spectrum roughness on the fuel and inner ablator surfaces. The 3D x-ray drive asymmetries from a 3D hohlraum simulation were incorporated into the radiation source.

The Miranda simulation is using  $512^3$  uniformly spaced grid points. To retain resolution of the capsule as it converges, the simulation grid is refined approximately every 50 ps during the course of the simulation. In this process the outer  $\sim 6\%$  of the domain is discarded and the problem is conservatively divided up onto a new  $512^3$  grid with increased resolution. In this way, the grid spacing starts at  $1.75\ \mu\text{m}$  and decreases to  $0.5\ \mu\text{m}$  by bang time. A simulation run on a  $1024^3$  grid without this refinement

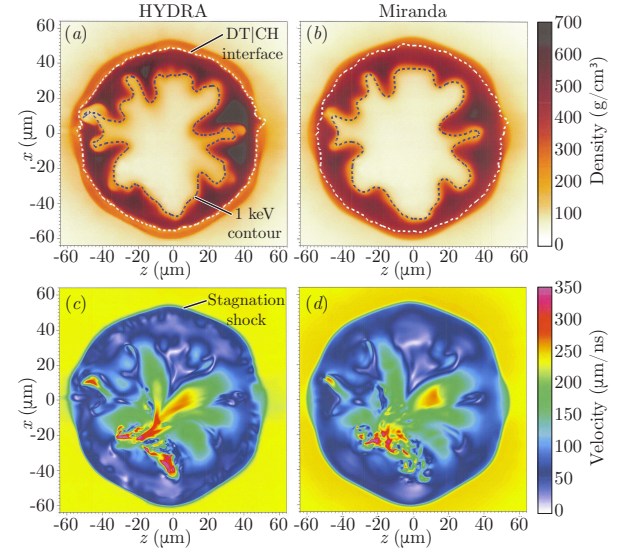


FIG. 2. Density (top) and velocity (bottom) slice from HYDRA (left) and Miranda (right) at bang time. The white contour on the density images shows the location of the fuel-ablator interface and the black contour shows the 1 keV boundary.

strategy showed nearly identical final results.

We first show that, without viscosity, both HYDRA and Miranda show a remarkably high level of agreement. This is notable given the complexity of the flow and the significantly different numerics of each code. By bang time (22.7 ns) the fuel-ablator interface has converged from a radius of  $340\ \mu\text{m}$  to  $54\ \mu\text{m}$ . Figure 2 shows density and velocity fields from a plane slicing through the capsule center. The Miranda simulation consists of approximately 256 grid cells across the width of the image, which is sufficient to resolve all pre-imposed perturbations. The HYDRA simulation's nonuniform mesh has approximately twice the resolution in the interior of the capsule compared to the Miranda simulation. The density field from the Miranda and HYDRA simulations show a very similar shape, with 10 individual high-density spikes penetrating towards the center of the capsule. The interface between DT and ablator material is displayed as a white dashed line, showing that perturbations on this interface do not inject ablator material into the hot spot. The hot spot, identified here as the 1 keV boundary and shown in Fig. 2 as a black dashed line, takes the shape of the inner surface of the DT fuel.

The lower half of Fig. 2 shows the velocity magnitude at bang time. Both HYDRA and Miranda show high-velocity features present in the hot spot. At upwards of  $350\ \mu\text{m/ns}$ , these low-density jets are comparable to the peak implosion velocity of the fuel. At this time, HYDRA and Miranda compute 0.71 kJ and 0.69 kJ of kinetic energy remains in the DT, respectively. These similarities lend confidence to the reliability of each model. The



question of whether higher resolution would produce increased hot spot turbulence, however, remains open. To this point, we next show that these simulations are effectively over-resolved due to the absence of physical viscosity modeling.

With the high temperatures in the hot spot, viscous effects become important at these small scales. To demonstrate this, an additional Miranda simulation is presented, identical to the previous case, except including a physical viscosity model. Figure 3 shows the  $4\pi$ -averaged profiles of several parameters at the time of peak fuel kinetic energy (22.41 ns). Notice that at this time the stagnation shock is entering the high-density fuel. Temperatures in the hot spot reach 2.3 keV. The thermodynamic regime of the plasma is governed by the plasma coupling parameter,  $\Gamma = (Z_i e)^2 / a k_B T$ , where  $Z_i e$  is the ionic charge,  $a$  is the ionic sphere radius, and  $k_B T$  is the ionic temperature. This parameter represents the ratio of potential energy from Coulombic interactions to the kinetic/thermal energy. When this value is very small, the viscosity can be described by kinetic theory [22]. Above the weakly coupled regime ( $\Gamma > 0.5$ ), molecular dynamics (MD) simulations are often relied upon to model the viscosity. Figure 3(b) shows that  $\Gamma$  is approximately 0.01 in the low density hot spot and 1-10 in the higher density DT fuel and CH ablator. The viscosity model implemented in Miranda for the current simulations uses the results of MD simulations of a Yukawa system [23] and corrected to asymptote to the kinetic regime at low  $\Gamma$  [22]. The kinematic viscosity, shown in Fig. 3(c), varies by 5 orders of magnitude from the hot spot to the ablation front.

The Reynolds number of the hot spot is given by  $Re = |\mathbf{u}'| L / \nu$ , where the fluctuating velocity is taken as the non-radial velocity,  $\mathbf{u}' = \mathbf{u} - U(r)$ , and  $U(r)$  is the  $4\pi$ -averaged radial velocity profile. For a length scale, the wavelength of mode 10 is used,  $L = 2\pi r / 10$ , as it is the dominant perturbation of the inner fuel surface at late times. The Reynolds number, shown in Fig. 3(c), ranges from 8 in the hot spot to  $2 \times 10^4$  near the fuel-ablator interface. This low Reynolds number implies that, even if these estimates of viscosity or length scale are in error by an order of magnitude, the viscous effects in the hot spot will still be considerable.

When viscous effects are included in the simulation, they act to dissipate small-scale velocity structures that are present in the hot spot. Figure 4(a) shows the non-radial velocity magnitude at 22.21 ns (0.2 ns before peak fuel kinetic energy) from the Miranda simulations with and without viscous effects. The dramatic smoothing of the velocity field in the presence of viscosity is evident.

The 3D turbulent kinetic energy spectra are shown in Fig. 4(b). These spectra use the fluctuating velocity fields and are computed within a box centered on the hot spot with sides that extend to the high-density fuel. The spectra are computed from both the viscous and inviscid cases

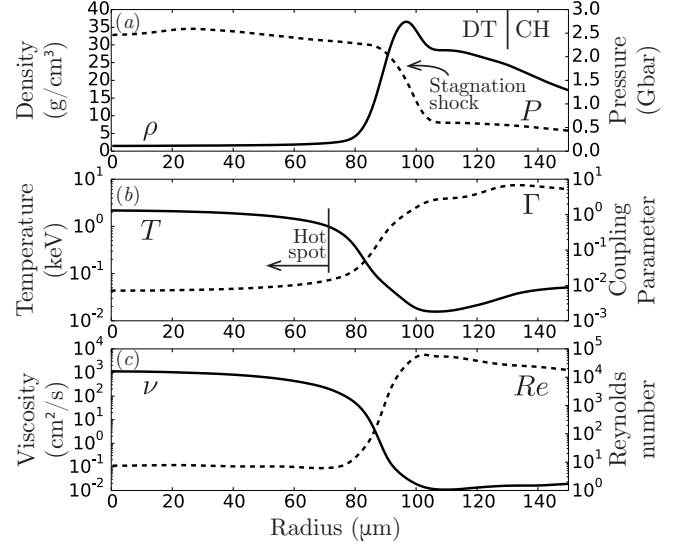


FIG. 3. Radial profiles, averaged over  $4\pi$ , at the time of peak fuel kinetic energy. (a) Density and pressure, (b) temperature and plasma coupling parameter, and (c) kinematic viscosity and Reynolds number. Solid curves correspond to the left axis, dashed curves correspond to the right axis.

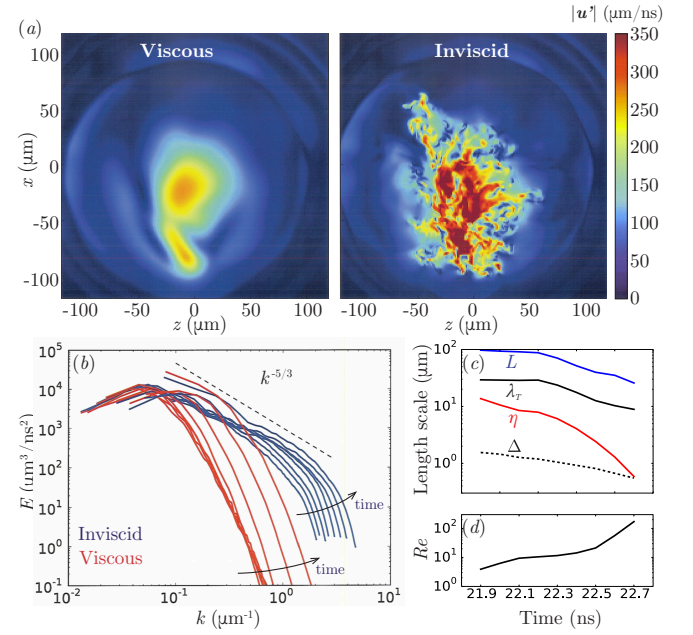


FIG. 4. Viscous effects on the fluctuating velocity. (a) Fluctuating velocity magnitude at 22.21 ns with (left) and without (right) viscosity. (b) Kinetic energy spectra in the hot spot from the viscous and inviscid simulations from 21.9 to 22.6 ns. (c) Turbulent length scales and (d) Reynolds number from the hot spot.

and at several times from 21.9 to 22.6 ns. The spectra show a much wider range of scales in the inviscid case, with the possible emergence of a turbulent  $k^{-5/3}$  inertial range [24] at the latest time. The high wavenumber region in the inviscid case shows the effects of artificial viscosity, which removes kinetic energy near the grid scale [19]. Physical viscosity, however, removes energy at much larger scales, as seen by the steeper drop-off in the viscous kinetic energy spectra.

Several characteristic length scales are extracted from the viscous simulation and are shown in Fig. 4(c). The largest is the integral length scale,  $L = \frac{3\pi}{2} \int k^{-1} E dk / \int E dk$ , where  $|\mathbf{u}'|^2 = \frac{3}{2} \int E dk$ , followed by the Taylor microscale,  $\lambda_T = (15\nu |\mathbf{u}'|^2 / \varepsilon)^{1/2}$ , where the dissipation is  $\varepsilon = \int 2\nu k^2 E dk$ , and finally the Kolmogorov scale,  $\eta = (\nu |\mathbf{u}'|^3 / \varepsilon)^{1/4}$  [24]. All length scales decrease in time due to the compression of the capsule. The Kolmogorov length scale, which is effectively the smallest scale of the flow, ranges from 10  $\mu\text{m}$  in the beginning of the simulation to 0.6  $\mu\text{m}$  by bang time. The grid spacing of the simulation,  $\Delta$ , is also shown in Fig. 4(c) and is much smaller than the Kolmogorov scale for most of the simulation. A resolution of  $\Delta \lesssim 2.1\eta$  is necessary to capture nearly all of the energy of the flow [24], therefore even at bang time, when the Kolmogorov and grid scales are approximately equivalent, all of the length scales in the hot spot are well resolved. The Reynolds number of the hot spot, computed here using the integral length scale and the rms velocity,  $|\mathbf{u}'|$ , is shown in Fig. 4(d). For much of the duration of the simulation it is on the order of  $Re \sim 10$ . The increase in  $Re$  near the end of the simulation ( $Re \approx 170$  at bang time) is due to the increase in the density of the hot spot, which reduces the kinematic viscosity, and the increase of non-radial kinetic energy in the hot spot during stagnation.

While viscosity has a significant effect on velocity fluctuations during the formation of the hot spot, the overall picture of the flowfield at bang time is not changed significantly. Figure 5(a) shows the density field from the viscous simulation with superposed velocity vectors. Compared with the inviscid simulation (Fig. 2(b)), viscosity slightly reduces the penetration of the fuel spikes into the hot spot, with some spikes reduced by  $\sim 5 \mu\text{m}$ . The overall concentration of mass, shown in Fig. 5(c) as fuel  $\rho R$  and ablator  $\rho R$ , is nearly identical for the inviscid and viscous case, as the two curves lie on top of each other. The fluctuating velocity fields, shown in Fig. 5(b), are also similar at bang time. The inviscid case shows smaller scale structures near the peak-velocity region, but outside this location the flowfields are nearly the same. Figure 5(d) shows the kinetic energy vs. time, divided up into the kinetic energy of all of the DT (top curve), the kinetic energy of the hot spot ( $T < 1 \text{ keV}$ , middle curve), and the turbulent (non-radial) kinetic energy (TKE) in the hot spot (bottom dashed curve). Be-

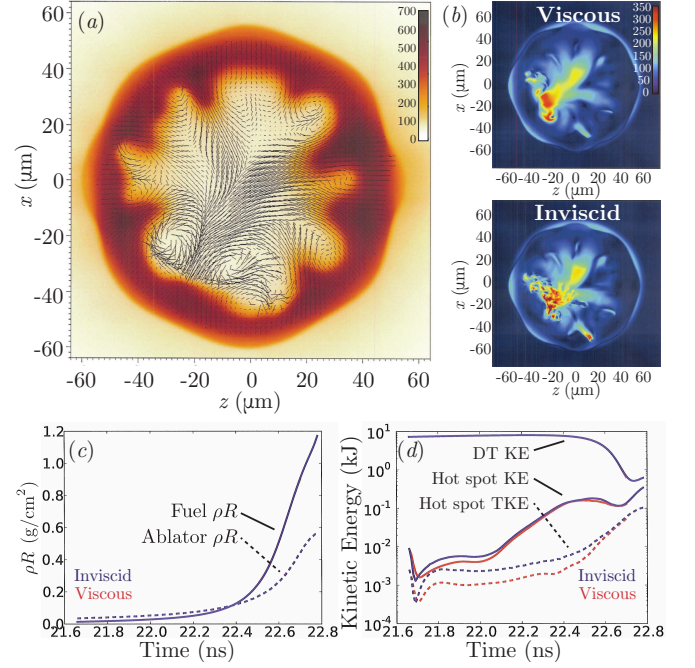


FIG. 5. Viscous effects at bang time on density, velocity, and integrated quantities. (a) Density and velocity vectors from the viscous simulation (vectors are only shown in the DT material for clarity). (b) Fluctuating velocity from the viscous (top) and inviscid (bottom) simulations. (c) Fuel and ablator  $\rho R$ . (d) Kinetic energy from all of the DT, from the hot spot, and turbulent (non-radial) kinetic energy in the hot spot. Each quantity in (c) and (d) has a viscous (red) and inviscid (blue) curves, but some viscous curves are hidden by the inviscid curves.

tween the start of the simulation and the time of peak fuel kinetic energy (22.41 ns), the viscous case has  $\sim 2\times$  less TKE as the inviscid case. After this period, both the inviscid and viscous cases experience a rapid increase in hot spot TKE, eventually reaching similar values at bang time.

The vector field in Fig. 5(a) demonstrates why both cases experience a similar increase in TKE near the end of the implosion. As the high density fuel stagnates and becomes Rayleigh-Taylor unstable, spikes fall into the hot spot and push jets of material through the interior. Figure 5(a) shows that the the direction of these jets are well correlated with the fuel spikes entering the hot spot. Thus there exists two sources of hot spot TKE. The first is the the initial shock waves, whose distortions lead to small-scale vorticity deposition. Viscous dissipation removes much of this kinetic energy. The second source of hot spot TKE is the inner surface of the high density fuel, which pushes material in front of it during stagnation. Having a low-mode shape, this inner surface produces low mode kinetic energy, which is not as affected by viscous dissipation.

This Letter shows that viscous dissipation prevents the formation of turbulence in the hot spot during ICF implosions. Viscosity strongly damps the small-scale velocity structure that is deposited by shocks reverberating in the interior of the capsule. Prior to fuel stagnation, the hot spot has a Reynolds number near 10. The additional energy and increase in density during stagnation raises the Reynolds number to near  $\sim 100$ , which is in contrast to the turbulent energy cascade seen in the inviscid simulation. The Kolmogorov length scale of 1-10  $\mu\text{m}$  indicates that ICF simulations with greater resolution than this will have an inaccurate picture of the flow if viscous effects are neglected. While the inclusion of viscous effects does not alter the averaged properties of the hot spot, the fine scale, detailed structure of the flow is significantly smoothed by the inclusion of physical viscosity.

We wish to thank Dr. W. H. Cabot for his work on modeling plasma properties in the Miranda code. This work was performed under the auspices of the U.S. Department of Energy by Lawrence Livermore National Laboratory under Contract No. DE-AC52-07NA27344.

---

\* weber30@llnl.gov

- [1] M. J. Edwards *et al.*, Phys. Plasmas **20**, 070501 (2013).
- [2] J. D. Lindl, P. Amendt, R. L. Berger, S. G. Glendinning, S. H. Glenzer, S. W. Haan, R. L. Kauffman, O. L. Landen, and L. J. Suter, Phys. Plasmas **11**, 339 (2004).
- [3] L. Rayleigh, Proc. R. Soc. London, Ser. A **14**, 170 (1883).
- [4] G. I. Taylor, Proc. R. Soc. London, Ser. A **201**, 192 (1950).
- [5] R. D. Richtmyer, Commun. Pure Appl. Math. **13**, 297 (1960).
- [6] E. E. Meshkov, Sov. Fluid Dyn. **4**, 151 (1969).
- [7] S. P. Regan *et al.*, Phys. Plasmas **19**, 056307 (2012).
- [8] T. Ma, P. K. Patel, N. Izumi, P. T. Springer, M. H. Key, L. J. Atherton, L. R. Benedetti, D. K. Bradley, D. A. Callahan, and P. M. Celliers, Phys. Rev. Lett. **111**, 085004 (2013).
- [9] D. S. Clark *et al.*, Phys. Plasmas **20**, 056318 (2013).
- [10] M. M. Marinak, G. D. Kerbel, N. A. Gentile, O. Jones, D. Munro, S. Pollaine, T. R. Dittrich, and S. W. Haan, Phys. Plasmas **8**, 2275 (2001).
- [11] H. F. Robey *et al.*, Phys. Plasmas **19**, 042706 (2012).
- [12] D. G. Hicks, N. B. Meezan, E. L. Dewald, A. J. Mackinnon, R. E. Olson, D. A. Callahan, T. Dppner, L. R. Benedetti, D. K. Bradley, and P. M. Celliers, Phys. Plasmas **19**, 122702 (2012).
- [13] S. H. Glenzer *et al.*, Phys. Plasmas **19**, 056318 (2012).
- [14] D. S. Clark *et al.*, in *Proceedings of the 8th International Conference on Inertial Fusion Sciences and Applications* (to be published).
- [15] S. W. Haan *et al.*, Phys. Plasmas **18**, 051001 (2011).
- [16] V. A. Thomas and R. J. Kares, Phys. Rev. Lett. **109**, 075004 (2012).
- [17] C. Cerjan, P. T. Springer, and S. M. Sepke, Phys. Plasmas **20**, 056319 (2013).
- [18] M. Gatu Johnson *et al.*, Phys. Plasmas **20**, 042707 (2013).
- [19] A. W. Cook, Phys. Fluids **19**, 055103 (2007).
- [20] A. W. Cook, Phys. Fluids **21**, 055109 (2009).
- [21] D. S. Clark, S. W. Haan, A. W. Cook, M. J. Edwards, B. A. Hammel, J. M. Koning, and M. M. Marinak, Phys. Plasmas **18**, 082701 (2011).
- [22] S. I. Braginskii, Reviews of Plasma Physics **1**, 205 (1965).
- [23] M. S. Murillo, High Energy Density Phys. **4**, 49 (2008).
- [24] S. Pope, *Turbulent flows* (Cambridge University Press, Cambridge, 2000).